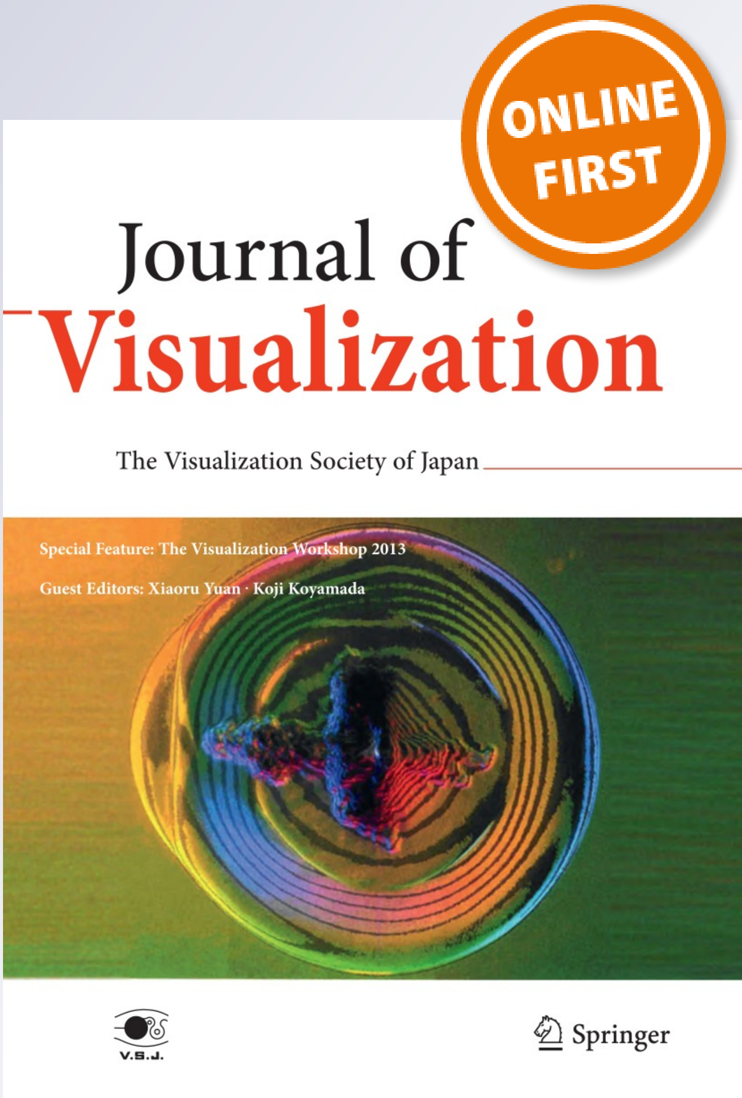
Visualization of flow instabilities in supersonic ejectors using Large Eddy Simulation

A. Bouhanguel, P. Desevaux & E. Gavignet

Journal of Visualization

ISSN 1343-8875

J Vis
DOI 10.1007/s12650-014-0231-4



Springer

Your article is protected by copyright and all rights are held exclusively by The Visualization Society of Japan. This e-offprint is for personal use only and shall not be self-archived in electronic repositories. If you wish to self-archive your article, please use the accepted manuscript version for posting on your own website. You may further deposit the accepted manuscript version in any repository, provided it is only made publicly available 12 months after official publication or later and provided acknowledgement is given to the original source of publication and a link is inserted to the published article on Springer's website. The link must be accompanied by the following text: "The final publication is available at link.springer.com”.

Springer

Author's personal copy

J Vis
DOI 10.1007/s12650-014-0231-4

SHORT PAPER

A. Bouhanguel · P. Desevaux · E. Gavignet

Visualization of flow instabilities in supersonic ejectors using Large Eddy Simulation

Received: 12 May 2014 / Revised: 7 July 2014 / Accepted: 14 July 2014
© The Visualization Society of Japan 2014

Keywords Ejector · Flow instabilities · LES simulation

1 Introduction

Ejectors are flow devices of rugged and simple construction, usually made up of two coaxial nozzles. They are used in a large variety of applications including vacuum pump, thermocompressor, thrust augmentation systems, hydrogen recirculation in fuel cell. However, ejectors involve very complex flows which have been the object of many numerical and experimental studies in the recent past years. These studies related mainly to the analysis of the shocks pressure recovery (Gaurav Singhal et al. 2010), the interaction and the mixing process between the primary and secondary flows (Yang et al. 2012) and the possible condensation which may occur in these supersonic flow devices. Many of the studies on ejectors have highlighted the relation between the ejector flow regime and the ejector performance in terms of vacuum and flow entrainment capacities (Hemidi et al. 2009). Knowledge of the flow pattern in ejectors is therefore essential and flow visualization techniques have been very useful to classify the flow in different flow regimes according to the choking or not of the flow in the secondary nozzle and to validate CFD simulations of the flow (Desevaux et al. 2013). Instabilities can also occur in ejector flows, being able to have a significant impact on the ejector performance, in particular on its mixing capacity (Murakami and Papamoschou 2001). These flow instabilities were sometimes visualized in experiments (Murakami and Papamoschou 2001), seldom numerically.

The objective of this paper is to propose a CFD simulation of the flow using the Large Eddy Simulation (LES) and to show its capability to numerically highlight the flow instabilities in a supersonic air–air ejector which were experimentally visualized in a previous work in our laboratory (Desevaux 1994).

2 Ejector configuration and CFD approach

The studied ejector is composed of two coaxial nozzles (Fig. 1). The primary nozzle (8-mm throat diameter, 12-mm exit diameter) is designed to provide an exit Mach number of 2.33. The secondary nozzle is made up of Altuglas permitting visualization of the flow. The mixing chamber of the ejector is modular in order to explore different flow regimes. Two diameters of the mixing chamber are used in this study, $D = 16$ mm and 24 mm, corresponding to secondary to primary throat area ratio $A = 4$ and 9, respectively. The length of the mixing chamber is 240 mm. Three-dimensional (3D) unsteady numerical simulations are performed

A. Bouhanguel · P. Desevaux (✉) · E. Gavignet
FEMTO-ST-UMR 6174, Département Energie, Université de Franche-Comté, 90000 Belfort, France
E-mail: philippe.desevaux@univ-fcomte.fr

Published online: 23 August 2014

Author's personal copy

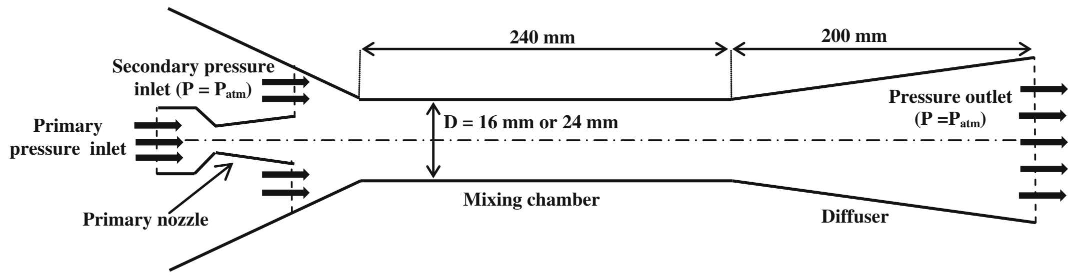


Fig. 1 Cross-sectional view of the ejector

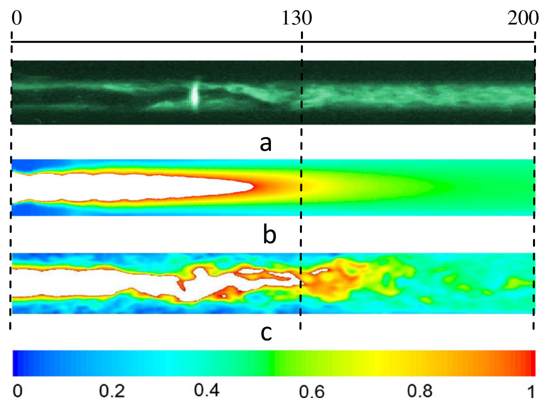


Fig. 2 Flow visualizations along the mixing tube in the mixed flow regime: **a** laser tomography, **b** subsonic iso-mach (RANS model), **c** subsonic iso-mach (LES model)

using ANSYS Fluent CFD software. A reduced computational domain (without settling chamber) is used to reduce calculation time. A grid mesh of 4 million fine tetrahedral cells with second order spatial and temporal discretization is necessary to lead simulations with time step of 1 µs for a total of 1 ms. As shown in Fig. 1, a stagnation pressure is fixed at the primary nozzle inlet while the atmospheric pressure is imposed at the ejector outlet and the secondary inlet. For the turbulence modeling, two approaches are used. The first one consists in using the k-omega SST RANS model which is suitable for supersonic flow with shocks (Kolar and Dvorak 2011). The second one uses the Large Eddy Simulation which consists in solving large turbulent eddies and modeling small structures of turbulence. The discretization scheme used is the Bounded Central Differencing with Smagorinksy-Lilly filtering algorithm.

3 Results and discussion

The results presented in this paper relate to two flow regimes within the ejector, obtained with two values of the geometrical ratio *A*. Figure 2 relates to the ejector of geometrical ratio $A = 9$, operating in a mixed flow regime. This flow regime is characterized by a supersonic jet at the primary nozzle exit and an induced flow which remains subsonic in the secondary nozzle. The flow field displayed in Fig. 2 covers the first 200 mm of the mixing tube length. Figure 2a presents a flow visualization obtained by pulsed laser tomography. This image shows flow instabilities occurring in the mixing tube during the interaction of the supersonic primary jet and the subsonic induced flow. Theses flow instabilities are significant, mainly in the region where the mixing process between the two flows is very active. Figure 2b and c show CFD visualizations of the subsonic iso-Mach lines. Thus, the supersonic flow appears in white color on these images. The unsteady CFD simulation performed using the RANS k-omega SST model (Fig. 2b) seems unable to predict appearance of these flow instabilities. On the contrary, the Large Eddy simulation (Fig. 2c) highlights the presence of fluctuations in the flow, particularly at the end of the supersonic jet. In this flow region, the mixing process, governed by free turbulent structures, is very active and the border between the supersonic jet and the subsonic flow becomes unstable. It is also observed that these fluctuations cause the disintegration of the supersonic jet in small regions of supersonic flow which are quickly dissipated in the flow

Author's personal copy

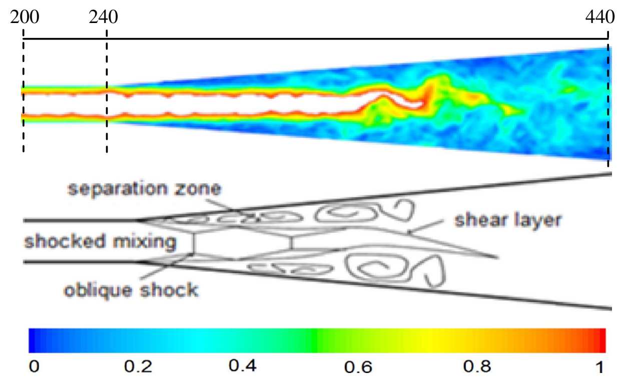


Fig. 3 Flow visualization along the diffuser in the supersonic flow regime: subsonic iso-mach (LES model) vs. explanatory scheme

before the end of the mixing tube. Figure 3 presents CFD visualization of the subsonic iso-Mach lines in the ejector diffuser. This result was obtained with the ejector of geometrical ratio $A = 4$, operating in fully supersonic regime. The flow made up of the mixture of the primary and secondary flows is supersonic along the totality of the mixing tube and decelerates to subsonic condition through a shock structure at the diffuser entrance. This simulation clearly shows the presence of fluctuations of the flow in the diffuser. Here too, these instabilities are concentrated at the end of the supersonic flow region. Indeed, the deceleration of flow from supersonic to subsonic causes flow separation within the boundary layer (Fig. 3) and the high shear stress on the supersonic jet induces its fluctuation.

This brief study shows the primacy of LES turbulence approach comparing to RANS turbulence models for the CFD simulation of instabilities in supersonic ejectors. These instabilities may appear when the supersonic jet is decelerated by the surrounding subsonic flow via a shear layer favoring the existence of a separation zone. This leads to fluctuations of the jet until the mixing is done. Only LES simulation succeeded in highlighting and visualizing these instabilities numerically. We are now working to improve the present CFD model using a finer mesh and smaller time step in the aim of investigating these flow instabilities more precisely. Quantitative experiments (flow visualization and image processing, 3D PIV) should complete this work and validate the LES simulations.

References

Desevaux P (1994) Analyse expérimentale du jet moteur d'un éjecteur à flux induit fonctionnant en régime mixte. J Phys III 4:1981–1988

Desevaux P, Bouhanguel A, Giradot L, Gavignet E (2013) On the use of laser tomography techniques for validating CFD simulations of the flow in supersonic ejectors. Int J Fluid Mech Res 40:60–70

Gaurav S, Mainuddin, Tyagi RK, Dawar AL and Subbarao PMV (2010) Pressure recovery studies on a supersonic COIL with central ejector configuration. Opt Laser Technol 42:1145–1153

Hemidi A, Henry F, Leclaire S, Seynhaeve JM, Bartosiewicz Y (2009) CFD analysis of a supersonic air ejector. Part II: relation between global operation and local flow features. Appl Therm Eng 29:2990–2998

Kolar J, Dvorak V (2011) Verification of k-omega SST turbulence model for supersonic internal flows. World Acad Sci Eng Technol 5:244–249

Murakami E, Papamoschou D(2001) Experiments on mixing enhancement in dual-stream jets. AIAA Paper 2001-0668

Yang X, Long X, Yao X (2012) Numerical investigation on the mixing process in a steam ejector with different nozzle structures. Int J Therm Sci 56:95–106